Reprint from proceedings of 15th National Conference on ... at R. E. C, Srinagar, ... July, ...

AERODYNAMIC DESIGN OF HIGH PRESSURE RATIO CENTRIFUGAL COMPRESSORS

By

S.Ramamurthy, S.Sankaranarayanan and K.Murugesan

Scientists, National Aeronautical Laboratory, Bangalore

ABSRTACT: This paper describes the aerodynamic design of high pressure ratio centrifugal compressors. The design approach is based on jet and wake model with suitable loss correlations in it. The method incorporates a matching diffuser design for optimum compressor performance. For a given input parameters like mass flow and pressure ratio, the computer programme developed based on the above method generates a 3-D compressor with optimised aerodynamic parameters. As an example the design of a high pr. ratio centrifugal compressor to develop a Pr. ratio of 6.0 at 0.96 Kg/Sec is discussed. This design method is one step improvement on the conventional design approach used widely in industries.

1. INTRODUCTION : Centrifugal compressors play an important role in small Aircraft engines, Turbochargers,Refrigeration plants, Chemical and Gas Industry. The design of these compressors rely heavily on empirical data derived from various publications. These empirical results deviate from fundamental fluid dynamic principles due to certain simplified physical assumptions. For example, ignorance of real flow phenomenon inside the blade passage, which is of major importance.

The internal flow in centrifugal compressor is highly complex in nature. Inside the blade passage, there exists a boundary layer growth in rotating coordinate system, with coriollis force acting on it. Flow near the compressor tip is almost always separated with a jet and a wake. At inducer tip and diffuser entry, the flow is supersonic with shock patches. No exact fluid dynamic model exists for these critical regions. The theoretical models like streamline curvature, Finite Difference and Finite element methods available in literature need modification to analyse the real flow behaviour in centrifugal compressors.

This paper describes a software tool for the design of centrifugal compressors. The design approach is based on jet and wake model which approximately simulate the real flow phenomenon at compressor outlet. For a specified parameters of mass flow rate and opressure ratio, the design of optimum aerodynamic configuration of the centrifugal compressor will be discussed.

As an example a high pressure ratio centrifugal compressor to develop a pressure ratio of 6 at 0.96 Kg/Sec was designed with back swept angle of 30 Deg.

The compressor inlet called inducer was designed for optimum specific speed and maximum mass flow per unit frontal area. Appropriate separation criteria and loss models were used to predict the flow at seperation point and at compressor outlet. A suitable model for mixing losses was used to predict the flow at diffuser inlet. The compressor flow passage was optimised for maximum efficiency. A matching diffuser was designed for this compressor using a simple design approach. This design method is one step improvement on the conventional design approach used widely in industries.

Notations:

m - Mass flow rate

N - Rotational speed

P - Pressure

Mr - Relative Mach number
a - Sonic velocity
γ - Ratio of specific heats
Q - Flow rate
Had- Adiabatic head
Hz - Number of compressor blades
W - Relative velocity
C - Absolute velocity
U - Blade speed
T - Temperature
Cp - Specific heat at constant pressure
Rc - Radius of curvature
r - Radius
D - Diameter
Ω - Angular velocity
ρ - Density of the fluid
λ_{2w} - Work done factor (Co2*/Cm2*)
β - Relative flow angle (w.r.t axial/radial)
α - Absolute flow angle (w.r.t axial/radial)
Prc- Pressure ratio total to static(P5/Pol)

Subscripts:

1 - Compressor inlet
2 - Compressor outlet
* - Diffuser inlet
4 - Throat
5 - Collector
s - Separation
j - Jet
w - Wake
Sh - Shear
o - Total
h - Hub
t - Tip
^ - Exponential
l - Limiting
g - Geometric

Note: All dimensions are in S.I units.

2. GOVERNING EQUATIONS:

A. Compressor Inlet (Inducer): The compressor inlet was optimised based on two parameters. One maximum mass flow per frontal area "F" (Ref-1), which is given by

$$F = \frac{4\pi m N^2}{3600 K P_{or} M_{r1t}\, a_{ot}}$$

$$= \frac{\left(\cos\beta_{1t} \cdot \sin\beta_{1t} + \cos\beta_{1t}\tan\alpha_{1t}\right)^2}{\left(1 + \frac{\gamma-1}{2}\frac{\cos^2\beta_{1t}}{\cos^2\alpha_{1t}} M_{r1t}^2\right)^{(3\gamma-1)/(2\gamma-2)}} \cdots$$

Where $K = 1-(D_{1h}/D_{1t})^2$

This equation relates mass flo with absolute and relative flow angle at inlet. Apart from inlet condition and geometric parameters, in no way the blade inlet angle or blade geometry is taken in. Hence, it is valid at all cases including optimum point of zero incidence. Figure-1 represents the above equation, which can be used to select an optimum eye proportion which corresponds to maximum value of F for a given limiting Mach number.

Specific speed is another parameter used to optimise the compressor inlet which is given by

Ns = N (Q) ^ 0.5 / ((Had) ^ 0.75 ..[2]

Centrifugal compressors as any class of turbomachinery show maximum efficiency at a particular specific speed around 38 metric units (Ref-2). To achieve this, N and Mrlt should be varied for a given mass flow and pressure ratio. However, the resulting Mrlt for maximum efficiency may not give rise to minimum frontal area through maximisation of the mass flow parameter function F. A compromise design will have to be chosen by iterating on speed and inlet relative tip Mach number.

With an assumed flow distribution at inlet the velocity triangles at hub, mean and tip sections can be generated. Solid body rotation or tan(Blt)/r=constant at compressor inlet give rise to a design with radial edge at inducer outlet to easily merge with the radial portion of the impeller (Ref-3). The axial length of the inducer should be fixed based on maximum diffusion and minimum frictional loss (Ref-4). The axial length of the inducer having 10 blades is represented in Figure-2 for various hub/tip ratio and relative flow angles at inlet. For other cases it is given by

$$\frac{\text{Axial Length}}{\text{Mean radius}} = \frac{10 * FX}{NZ} \quad .. \quad [3]$$

Where FX is the axial length of inducer having ten blades.

A shockless inflow requires a positive incidence, which is calculated from the blockage factor due to blade thickness.

B. Separation Point: One dimensional design assumes that flow through the compressor passage is full with constant diffusion from inlet to outlet. The diffusion ratio in this case is the ratio of relative velocity at inlet and at outlet and is termed as geometrical diffusion ratio DRg.

$$DRg = W1t/W2 \quad .. \quad [4]$$

In real flow due to complex blade length associated with coriollis force in rotational frame, flow tries to separate in the corner of the tip shroud and blade suction surface as shown in Figure-3. The diffusion ratio in this case is the ratio of relative velocity at inlet and at separation point. It is termed as limiting diffusion ratio DRl.

$$DRl = W1t/Ws \quad .. \quad [5]$$

Flow is assumed to separate (Ref-5) when DRg exceeds DRl. The properties at separation point is calculated using conservation of relative total enthalpy and an assumed Drl commensurate with the inducer design.

$$Ts = T1 + \frac{W1t\wedge 2 - U1t\wedge 2}{2Cp} - \frac{Ws\wedge 2 - Us\wedge 2}{2Cp} \quad [6]$$

$$Ps = (Ts/T1)\wedge(\gamma/(\gamma-1)) - \Delta pd \quad .. \quad [7]$$

Where pd = Diffusion loss in the inducer.

C. Compressor outlet: From the separation point the wake starts growing on the suction surface with jet and wake at compressor outlet as shown in Figure-4. In the jet the flow is assumed to be isentropic and at constant relative Mach number.

The flow at compressor outlet is defined by three non dimensional parameters.

$$\lambda = \frac{\text{Wake mass flow rate}}{\text{Total mass flow rate}}$$

$$\nu = \frac{\text{Relative velocity in the wake}}{\text{Relative velocity in the jet}} \quad [8]$$

$$k = \frac{\text{Wake width}}{\text{Blade pitch}}$$

The jet and wake is a conceptual model by hypothesizing the flow separation inside the compressor. In this model these two regions are considered to be adjoining each other and flowing without any inter mixing within the compressor. Hence it is only correct to assume that their relative flow angles are the same with a shear layer in between.

$$\text{i.e.,} \ \beta_{2j} = \beta_{2w} = \beta_2 \quad .. \quad [9]$$

Static pressure in the wake and jet different. This is balanced by the equilibrium of centrifugal and coriollis force without any cross flow inbetween. The static pressure in the wake is given by

$$P_{2w} = P_{2sh} - \frac{0.5\pi D_2 \epsilon \rho_{2w} \cos\beta_2}{N_z}\left(2\Omega W_{2w} - \frac{W_{2w}^2}{R_{ow}}\right) \quad ..(10)$$

Where

$$P_{2sh} = P_{2j} - \frac{0.5(1-\epsilon)\pi D_2 \rho_{2j} \cos\beta_2 \ 2\Omega W_{2j}}{N_z} - \frac{W_{2j}^2}{R_{oj}}$$

The unknown 2 is calculated from the overall slip factor is given by

$$\mu = (\cos(\beta 2b)\wedge 0.5)/(Nz\wedge 0.7) \quad .. \quad [11]$$

Impeller outlet width is calculated by satisfying mass flow through the jet.

$$b2 = (1-\lambda)*m/(PI*D2*\rho 2j*W2j*BK*(1-\epsilon)*\cos\beta 2) \quad .. \quad [12]$$

Where BK=Blockage factor due to blade thickness.

The equivalent condition at compressor outlet is calculated by mass averaging the properties of jet and wake. Various loss parameters like Frictional loss, Leakage loss, Back flow loss and Disk friction loss are calculated as explained in Reference-7 using velocity vector diagrams and compressor geometry. These losses were added to equivalent temperature at compressor outlet to get effective total temperature at diffuser inlet.

D. Vaneless Diffuser: The wake and the jet at compressor outlet is assumed to mix out into a uniform flow as the flow reaches the diffuser inlet. During mixing process there exists (a) Mixing loss because of energy exchange between jet and wake due to pressure difference (b) Frictional loss in the diffuser walls. These losses were calculated using the method given by Reference-8.

The mixing loss due to sudden expansion in the vaneless portion is given by

$$\omega_M = \frac{1}{1+\lambda_{2*}}\left(\frac{1-\epsilon-B}{1-\epsilon}\right) \quad \ldots (13)$$

Where B = (Width of the vaneless portion/ Width of compressor outlet)

The frictional loss in the vaneless portion is given by

$$\omega_F = 1 - Cp - ((C/C2)\wedge 2) \quad .. \quad [14]$$

The effective total pressure at diffuser inlet is calculated by subtracting the mixing and frictional loss in the vaneless portion from the equivalent total pressure at compressor outlet. At this stage total to total efficiency can be predicted from compressor inlet condition and the mixed out condition at compressor outlet.

$$\eta_{T-T} = \frac{T_{01}\left[\left(\frac{P_{02*}}{P_{01}}\right)^{\frac{\gamma-1}{\gamma}} - 1\right]}{\left(T_{02**} - T_{01}\right)} \quad \ldots (15)$$

Where To2** = T02* + (Disc Friction + Clearance + Back Flow)loss

The combined velocity vector diagrams at compressor outlet and at diffuser inlet is represented as shown in Figure-5. The typical T-S diagram with associated losses are shown in Figure-6.

E. Vaned Diffuser: For an assumed throat Mach number the pressure recovery from diffuser inlet to throat and blockage due to boundary layer are calculated from Figure-7 (Reference-5).

$$Cp2*,4 = (P4-P2*)/(Po2*-P2*) \quad .. \quad [16]$$

From the above values and mass flow rate throat dimension is estimated. Assuming the diffuser after the throat as a simple two dimensional diverging channel, the overall dimensions of the diffuser are estimated for an assumed divergence angle and number of blades. The overall pressure recovery and total to static efficiency of the stage are calculated by estimating the loss in the diffuser.

$$Cpd = (P5-P2*)/(Po2*-P2*) \quad .. \quad [17]$$

$$\eta_{T-S} = \frac{\left[\left(\frac{P_S}{P_{01}}\right)^{\frac{\gamma-1}{\gamma}} - 1\right] T_{01}}{\left(T_{02**} - T_{01}\right)} \quad \ldots (18)$$

3. RESULTS AND DISCUSSIONS: A computer programme was developed based on the method described in the previous section to design centrifugal compressors. The compressor design follows step by step as indicated in the flow chart (Figure-8). The design method is based on iterative procedure with three aspects. They are (a) Design of compressor inlet for optimum specific speed (b) Design of compressor outlet to get the required pressure ratio with minimum losses and (c) Design of diffuser for maximum pressure recovery and overall efficiency. As a demonstration of the method a high pressure ratio centrifugal compressor was designed to develop a pressure ratio of 6.0 at a mass flow rate of 0.96 Kg/Sec. with a back swept angle of 30.0 Deg.

The inducer inlet was designed by selecting various combinations of rotational speed, relative tip Mach number and relative flow angle. Figure-9 shows the variation of these parameters and chosen design point is also indicated therein. In order to satisfy the mass flow rate and pressure ratio requirement and specific speed to be close to optimum with reasonable hub/tip ratio the rotational speed and relative tip Mach number should be around 65,000 and 1.09 respectively. With the inlet geometry being defined the velocity vector diagrams (Figure-10) at hub, mean and tip sections were calculated assuming the flow to behave like a solid body rotation. Circular arc camber was assumed for the inducer blade section from the consideration of feeding the blade co-ordinates to numerically controlled machine for milling the compressor. The blade thickness was assumed to vary linearly from hub to tip. The inducer length was fixed from the consideration of maximum diffusion with minimum frictional loss.

The blockage factor at inducer inlet was estimated. The estimated incidence angle at inducer tip for shockless inflow was around 3.6 Deg. For this incidence angle and Mrlt=1.09 the diffusion ratio as suggested by Dean (Reference-9) is around 1.425 which is a typical value for high pressure ratio compressors of the type considered here. With the assumption that DRg exceeds 1.425, the flow is assumed to separate at the exit of the inducer. The actual static pressure at the separation point was calculated by estimating the diffusion loss in the inducer.

The properties of jet and wake at compressor outlet were calculated separately with the different criteria. In the jet it was assumed that (a) Flow is isentropic (b) Relative Mach number remains constant. In the wake the following conditions were satisfied. (a) Assumed mass flow through the wake. (b) The pressure balance between the jet and the wake along the shear layer. The flow angle at the compressor outlet was calculated using an iterative procedure by satisfying the overall slip factor and $\beta_{2j} = \beta_{2w}$. The various geometric parameters derived from computer programme are used to generate blade co-ordinates for machining. The equivalent velocity vector diagram at diffuser inlet was obtained by allowing the jet and the wake to mix out into a uniform stream. The velocity vector diagrams at the compressor outlet and at the diffuser inlet are shown in the Figure-10. The diffuser throat Mach number was fixed at 0.95. The diffuser after the throat was designed based on simple two dimensional channel with constant divergence angle equal to 10 Deg. The diffuser outlet Mach number obtained by an iterative solution is around 0.13.

From the overall geometry of the compressor the blade co-ordinates were generated by assuming suitable shapes for hub, shroud contour and blade surface. These co-ordinates were converted into isometric co-ordinates by matrix transformation. The isometric view of the compressor generated using these co-ordinates is shown in Figure-11.

4. CONCLUSIONS:

The method developed for the Aerodynamic Design of Centrifugal Compressors is based on Jet and Wake model with suitable loss correlations and empirisms in it. The method is one step improvement on one dimensional approach used widely in industries. This design software is useful in generating co-ordinates at close intervals, which can be fed directly to N/C machine processor for milling the compressor. This design software is also useful to visualize and modify the flow channel on the computer CRT screen. As a first step this design software can be used as preliminary tool for aerodynamic design and performance evaluation of centrifugal compressor used widely in industries .

5. REFERENCES:

(1) Shepherd.D.G, Principles of Turbomachinery. The Macmillan Company, New York.

(2) Balji.O.E., A study on design criteria and matching of turbomachines. Part B. Compressors and pump performance and matching of turbo components: ASME paper 60-WA-231, 1960.

(3) On the unresolved fluid dynamics of the centrifugal compressors. VKILS 25, 1970 and VKILS 50, 1972.

(4) K.Bammert and A.Mobarak, Flow studies in the inducer part of a centrifugal compressor, 20th annual Int. Gas Turbine Conference of the ASME Houston/Texas USA March 1975.

(5) Dean.R.C. Jr., The Fluid Dynamic Design of Advanced Centrifugal Compressors. TN-153, Sept. 1972 Creare Inc.

(6) Wiesner.F., A review of slip factors for centrifugal compressors. ASME Trans., Series: A; Journal of Engg. for power, Vol.89, No.3, October 1977, P558.

(7) Jansen.W, A method for calculating the flow in a centrifugal impeller when entropy gradients are present.

(8) Johnston.J.P. and Dean.R.C. Jr., Losses in Vaneless Diffusers of Centrifugal Compressors and Pumps. Analysis, Experimental and Design. Trans. of ASME Journal of Engg. for power, Jan 1966.,PP:49-62.

(9) Osborne.C, Runstadler.P.W and Dodd Stacy.W, Aerodynamic and Mechanical Design of an 8:1 pressure ratio centrifugal compressor., Creare TN-204.

6. ACKNOWLEDGEMENT: The authors wish to thank Mr. R.Rajendran of Propulsion Division for the technical help given in preparing this manuscript.

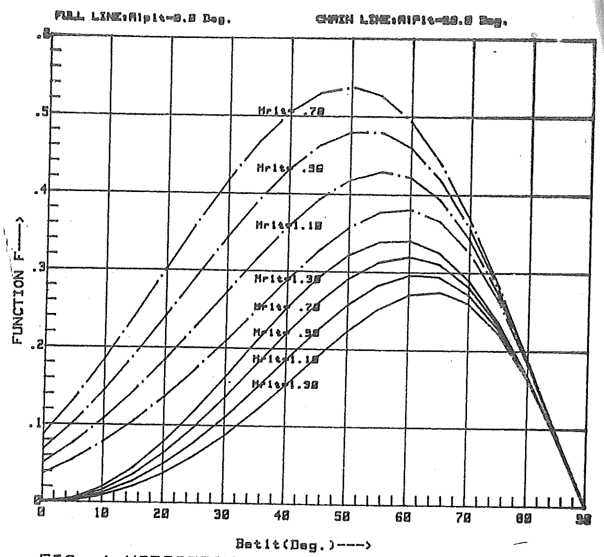

FIG: 1 VARIATION OF F WITH INLET FLOW ANGLE

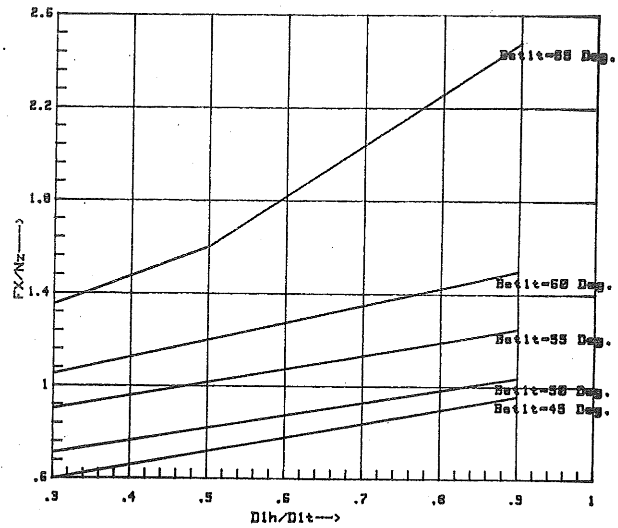

FIG: 2 AXIAL LENGTH OF INDUCER

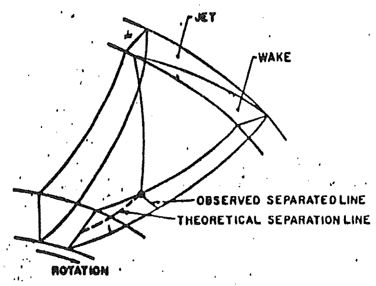

FIG. 3 SCHEMATIC OF SEPARATION LINE GEOMETRY

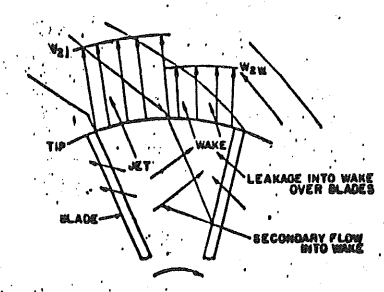

FIG. 4 SCHEMATIC OF SEPARATED FLOW MODEL

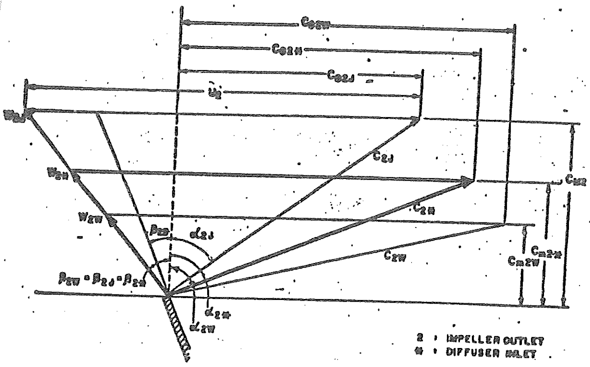

FIG. 5 OUTLET VELOCITY TRIANGLE

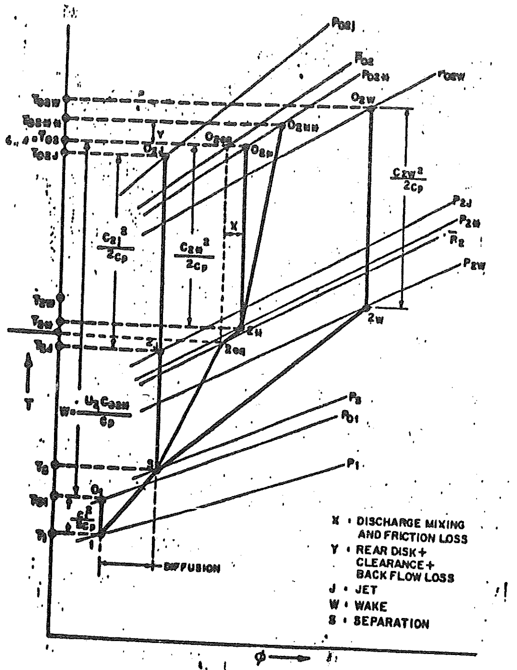

FIG. 6 T-φ DIAGRAM

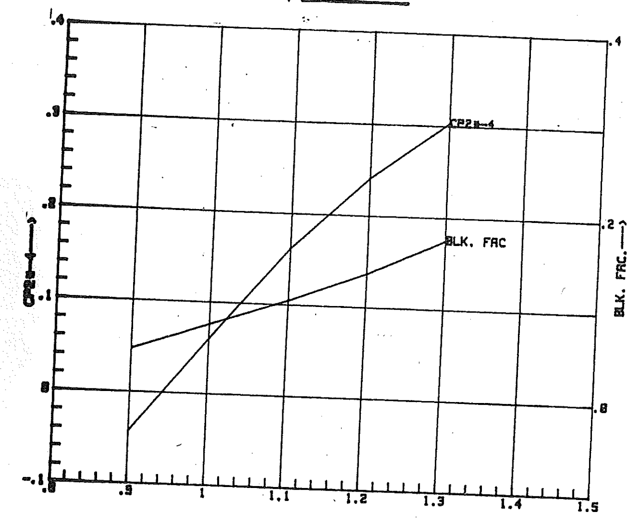

FIG:7 VARIATION OF CP2*-4 & BLK. FRC. WITH M2*(Ref-5)

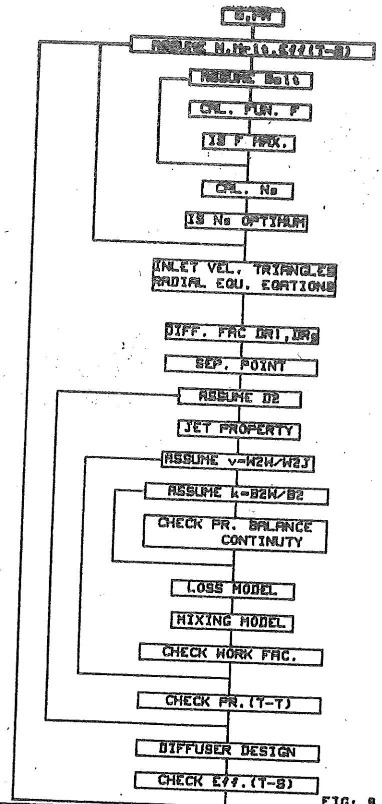

FIG: 8 FLOW CHART

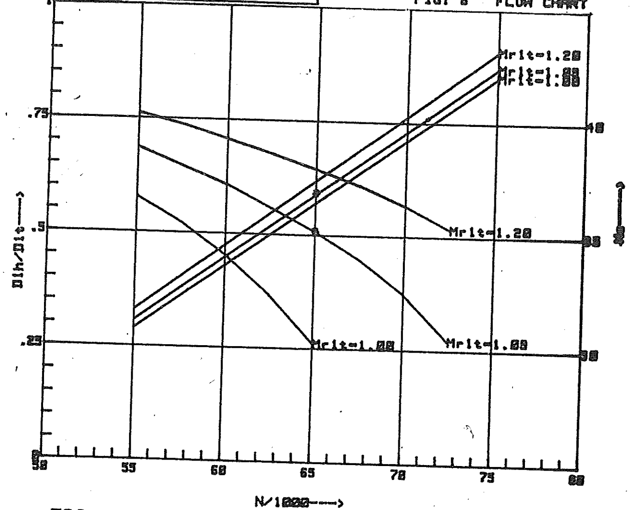

FIG: 9 VARIATION OF Ns AND D1h/D1t WITH N

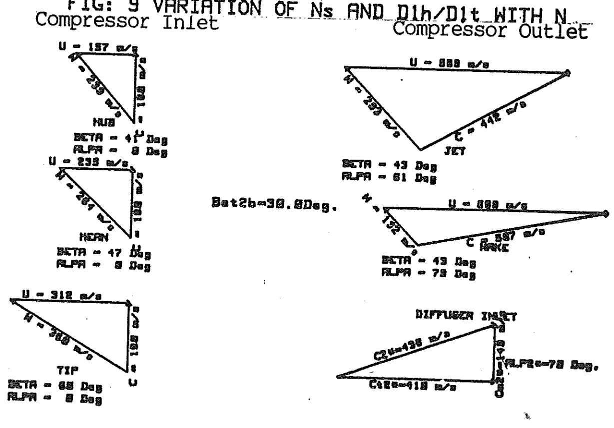

FIG:10 VELOCITY VECTOR DIAGRAMS